Numerical modelling of pressure piling with inhomogeneous hydrogen/air mixtures

The design of electric valve actuators for Ex-d hazardous environments in which an explosive event must be contained is examined in this study[1]. Combustion within these small enclosed volumes is complicated by the irregular geometries developed in the design process (effectively a series of baffles). These baffles lead to a scenario whereby combustion within the enclosed interconnected vessels produces a phenomenon called 'pressure piling' which usually results in deflagration to detonation transition (DDT).

Pressure piling has an important role in certification testing of commercial actuators, as it can produce up to eight times the standard combustion pressures causing design failures and ultimately a non-certifiable product. It is caused by unburnt gases ahead of a flame front compressing and heating up causing one large explosion.

Pressure piling has not been numerically studied extensively due to the niche nature of the problem, however, some simulations have produced comparative results with experiments using homogenous mixtures [2]. The present study aims to investigate pressure piling within inhomogeneous mixtures to identify the mechanisms which accelerate and decelerate the pressure rates producing the large over-pressures. This includes varying the geometry, such as blockage ratio, and initial conditions, such as ignition position to measure the affects. Inhomogeneous mixtures within conventional obstructed channels have been seen to produce large over-pressures [3] but not within a geometry that can produce pressure piling.

Two simple cylindrical vessels connected by a transmission pipe with the first primary vessel having a larger volume than the second vessel (replicating Singh's experiments for verification [1]), are presented here to induce the pressure piling phenomenon. Models for pressure calculation based on experimental results from experts in this field are presented to verify the accuracy of the numerical solver.

OpenFOAM has been employed for CFD simulations which is a C++ based, open source toolbox for the development of customized numerical solvers, and pre-/post- processing utilities for the solution of continuum mechanics problems, including combustion [4]. "*rhoReactingFoam*" [3] has been thoroughly examined and utilised, it is a density-based solver with chemical reactions for combustion and suitable for sub- and supersonic fluid velocities. Simulation results will be compared to the results of Singh's experiments to validate the solver [1]. A further complex experiment which includes obstacles (i.e. blockage ratios) are planned for July 2018 to further verify the accuracy of the numerical solver.

The expected results will show the impact of the factors affecting pressure piling including initial conditions such as: high temperature, turbulent velocity field, high pressure, inhomogeneous mixtures (concentration gradients), ignition position, volume ratio (between the two vessels) and geometry (venture tube as interconnecting tube for example). This could prove that inhomogeneous concentration gradients have the largest effect on the over-pressure produced. Eventually, this will lead to design techniques to control the effects of pressure piling.

Future work includes integrating the verified solver into a fluid-solid interactions (FSI) solver which will measure the effects on flame propagation and maximum pressures produced due to the deformation of the outer body.

References:
[1] Singh, J. (1994). Gas explosions in inter-connected vessels: pressure piling. Transactions of the IChemE, 72B, 220e228.
[2] E. Salzano A. Di Benedetto. “CFD simulation of pressure piling”. In: *Journal of Loss Prevention in the Process Industries* 23 (2010), pp. 498–506.
[3] Reza Khodadadi Azadboni Jennifer X. Wen Ali Heidari Changjian Wang. “Numerical modelling of deflagration to detonation transition in inhomogeneous hydrogen/air mixtures”. In: *Journal of Loss Prevention in the Process Industries* (Apr. 2017).
[4] OpenFOAM Ltd, 2015. OpenFOAM. Available from: http://www.openfoam.com/.
[5] Rotork Controls Ltd. *Rotork Controls Ltd.: Controlling Flow*. URL: http://www. rotork.com/en/.

[1] This work is being carried out in conjunction with Rotork Controls Ltd, Bath, UK [5].